Copy No. _____

Defence Research and Development Canada
Recherche et développement pour la défense Canada

DEFENCE R&D DÉFENSE

Virtual Crack Extension Technique

Khaled Shahin
Tamunoiyala Koko
Martec Limited

Martec Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia B3J 3J8

08.410438
Contract Project Manager: Dr. T. Koko
Contract Number: W7707-088096/A
Contract Scientific Authority: Christopher Bayley, 250-363-4784

Defence R&D Canada – Atlantic

Contract Report
DRDC Atlantic CR 2008-274
December 2008

Canada

This page intentionally left blank.

Virtual Crack Extension Technique

Khaled Shahin
Tamunoiyala Koko
Martec Limited

Prepared By:
Martec Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia
Canada B3J 3J8

08.410438
Contract Project Manager: Dr. T. Koko
Project Number: W7707-088096/A
Contract Scientific Authority: Christopher Bayley, 250-363-4784

Defence R&D Canada – Atlantic

Contract Report
DRDC Atlantic CR 2008-274
December 2008

Principal Author

Original signed by Dr. Khaled Shahin

Dr. Khaled Shahin

Research Engineer

Approved by

Original signed by Dr. Terry Foster

Dr. Terry Foster

Section Head DLP

Approved for release by

Original signed by Ron Kuwahara for

Dr. Calvin Hyatt

Chair DRP

Abstract

In order to numerically calculate the strain energy release rate (SERR) using finite element models, an implementation of the virtual crack extension technique was developed within a Matlab script. From the nodal data read in from the ascii output files produced by LS-Dyna, the SERR was calculated from contours of elements which were positioned around the crack tip. In order to validate the implementation, a mesh sensitivity study of a center cracked panel was carried out. This study concluded that the SERR implementation was relatively independent of mesh size and shape.

Résumé

Nous avons intégré la technique d'extension virtuelle des fissures dans un script Matlab, afin de calculer le taux de dissipation de l'énergie de déformation, à l'aide d'un modèle d'éléments finis. À partir des données nodales tirées des fichiers de sortie ASCII du logiciel LS-Dyna, nous avons calculé le taux de dissipation d'énergie de déformation pour les contours des éléments disposés autour de l'extrémité de la fissure. Cette intégration a été validée par une étude de sensibilité pour le maillage d'un panneau fissuré au centre. La présente étude a démontré que notre intégration du calcul de dissipation de l'énergie de déformation était relativement indépendante de la taille et de la forme des mailles.

This page intentionally left blank.

Executive summary

Virtual Crack Extension Technique

K. Shahin; T.S. Koko; DRDC Atlantic CR 2008-274; Defence R&D Canada – Atlantic; December 2008.

Introduction or background: This report describes the derivation and development of a Matlab program that computes the strain energy release rate (SERR) in two dimensional finite element models. The program was developed to post-process displacement results from LS-Dyna using the continuum mechanics formulation of the virtual crack extension technique (VCET). Brief descriptions of the VCET and other methods of extracting SERR results are summarized as well.

Results: The SERR for a center crack panel are computed for various FEM mesh designs and compared with analytically derived formulas. The examples illustrate that the derived SERR formulations is correctly implemented for plane stress models.

Significance: These SERR post-processing routines allow fracture mechanics variables to be computed directly from numerical analyses. By comparing the computed SERR to those acquired experimentally, the onset of fracture can be estimated directly from the simulation.

Future plans: Future plans include the extension of the SERR technique to include elastic-plastic material behaviour and solid elements. This will allow a variety of geometries and materials to be modelled.

Sommaire

La technique d’extension virtuelle des fissures

K. Shahin; T.S. Koko; DRDC Atlantic CR 2008-274; R & D pour la défense Canada – Atlantique; Décembre 2008.

Introduction ou contexte : Dans ce rapport, nous décrivons la production d’un script Matlab pour le calcul du taux de dissipation de l’énergie de déformation pour les modèles bidimensionnels d’éléments finis. Ce programme applique la technique d’extension virtuelle des fissures avec le formalisme de la mécanique des milieux continus, sur les résultats de déplacements calculés avec le logiciel LS-Dyna. Nous décrivons brièvement la technique d’extension virtuelle des fissures et d’autres méthodes d’extraction des résultats du taux de dissipation de l’énergie de déformation.

Résultats : Nous avons calculé le taux de dissipation de l’énergie de déformation pour différents schémas du maillage du modèle d’éléments finis et les avons comparés avec les formules dérivées analytiquement. Ces exemples montrent que les formulations dérivées de la dissipation de l’énergie de déformation sont correctement intégrées aux modèles de contraintes dans les structures planes.

Importance : Ces routines post-traitement de dérivation de la dissipation de l’énergie de déformation permettent de calculer directement les variables de fracture mécanique à partir des analyses numériques. On peut estimer directement le point de déclenchement de fracture, en comparant les taux de dissipation de l’énergie de déformation, aux valeurs expérimentales.

Perspectives : Les recherches futures porteront sur le développement de la technique de calcul du taux de dissipation de l’énergie de déformation pour y inclure le comportement des matériaux élastiques-plastiques et les éléments solides, ce qui permettra de modéliser diverses géométries et substances.

Table of contents

List of figures

List of tables

This page intentionally left blank.

1 Finite Element Formulation

The total strain energy release rate (SERR) using the virtual crack extension method is given by Equation 1 for the case of a cracked structure with no body force component and in the absence traction forces at the crack face.

$$G_T = \frac{1}{A_c}\int_V \left\{ \left(\sigma_{ij}\frac{\partial u_j}{\partial x_k} - W\delta_{ik} \right) \frac{\partial \Delta x_k}{\partial x_i} \right\} dV \qquad (1)$$

where,

G_T = total strain energy release rate,
A_c = area of crack extension,
σ_{ij} = stress tensor,
u_j = nodal displacement,
W = strain energy density,
δ_{ik} = Kronecker's delta, and
Δ_x = nodal virtual extension,

Using Gauss Quadrature integration method, the SERR is obtained using the following expression to approximate the volume integral given by Equation 1 for two-dimensional plane stress or plane strain conditions:

$$G_T = \frac{1}{\Delta a}\sum_{l=1}^{Ne}\sum_{i=1}^{n}\sum_{j=1}^{m} trace\left[\left(\{\sigma\}\left\{\frac{dU}{dX}\right\} - W\{I\} \right) \left\{\frac{d\Delta X}{dX}\right\} \right] \alpha_i \alpha_j \det J \qquad (2)$$

where,

Δa = virtual crack extension,
N_e = number of elements in the model,
n, m = order of Gauss Quadrature integration,

$$\left\{\frac{dU}{dX}\right\} = \{U_o\}[P][J]^{-1}$$

$$\left\{\frac{d\Delta X}{dX}\right\} = \{\Delta X_o\}[P][J]^{-1}$$

$[I]$ = 2x2 identity matrix,
$\{U_o\}$ = nodal displacement vector,
$\{\Delta X_o\}$ = virtual extension of element nodes,
[P] = derivatives of shape functions,
[J] = element Jacobian, and
α = Gauss quadrature weight factors.

See deLorenzi (1985) for more detailed description of the expressions shown above and their derivations.

Instead of relying on stress and strain outputs from LS-Dyna, to both reduce the amount of manual pre-processing required and to maintain consistency in the formulation, the required stress and strains are derived from LS-Dyna displacement results as given below

$$\{\sigma\} = [D]\{\varepsilon\}$$
$$\{\varepsilon\} = \begin{Bmatrix} \frac{dU}{dX}(1,1) \\ \frac{dU}{dX}(2,2) \\ \frac{dU}{dX}(1,2) + \frac{dU}{dX}(2,1) \end{Bmatrix} \tag{3}$$

Therefore, nodal displacement results are the only required output from LS-Dyna, and only for the elements involved in the SERR calculations. Furthermore, the SERR contribution of elements in which all nodes undergo the same (or no) crack extension is exactly zero. Gauss quadrature integration is performed using one-point rule, in consistency with the LS-Dyna reduced integration of the element stiffness matrix.

2 Procedure:

To use the ***vcet_martec.m*** program, the following procedure is recommended.

Define the elements surrounding the crack under a unique part ID (component) separate from the rest of elements in the model. These elements will be used in the calculation of the SERR in the ***vcet_martec.m*** program.

For convenience, re-number the nodes and elements in this component to start at 1, and the rest of the elements and nodes in the model to start a much larger number. This is not strictly necessary, however, it greatly improves the experience of specifying the nodal shifting to be specified by the user and the process of obtaining element nodal coordinates and displacements.

The direct input to the program is done through the text file **input_parameters.dat**, which contains the following comma separated input parameters:

file1.csv, file2.csv, file3.csv, file4.csv, psID, symmetryID, cwID, modulus, pr, delta_x

where
file1.csv : contains the node coordinates input (character string)
file2.csv : contains the element nodal connectivity input (character string)
file3.csv : contains the nodal displacement solution (character string)
file4.csv : contains the nodes affected by virtual crack extension (character string)

psID:	Element formulation option: (integer) EQ. 12: Plane stress (default) EQ. 13: Plane strain
symmetryID:	Model symmetry option (integer) EQ. 0: Not symmetric (default) EQ. 1: Crack lies on plane of symmetry.
cwID :	Element node numbering order (integer) EQ. 0: Counter clockwise (default) EQ. 1: Clockwise.
modulus :	Elements Young's modulus (float)
pr :	Poisson ratio (float)
delta_x :	Virtual crack extension (float)

Example of input line (used in verification example 1):

c_nodes.csv, c_elements.csv, c_disp.csv, c_paths.csv, 12, 1, 1, 207000, 0.3, 0.001

Note: character-type (file name) input is case sensitive.

As mentioned above, the ***vcet_martec.m*** program requires four input files. Below, a brief description is given on the format and input parameters of the four files.

1. **file1.csv:** nodal coordinates input
   From the LS-Dyna input file, copy the nodal coordinates of the nodes attached to the crack-elements component into Excel and remove the column corresponding to the z-coordinate. Save this information in a comma separated ".csv" format.

   Input format: Node number, x-coordinate, y-coordinate

2. **file2.csv:** element-node connectivity input
   Obtain a list of elements nodal connectivity from the LS-Dyna input file. Copy this information into Excel **and remove the column corresponding to the Part ID Number** (second column in LS-Dyna input). Save this information in comma separated ".csv" format.

   Input format: Element number, node 1, node 2, node 3, node 4

3. **file3.csv:** node displacement solution
   Copy the nodal displacement solution obtained from LS-Dyna "**nodeout**" ASCII file into Excel, and save only the columns corresponding to the node numbers, and displacements in the x- and y- directions.

   Input format: node number, x-displacement, y-displacement

4. **file4.csv:** nodes affected by virtual crack extension
   Specify the nodes affected by the virtual crack extension. Obtain the node numbers from LS-Dyna input file, list them in Excel such that each column contains the nodes to be shifted in each path, and save in comma separated ".csv" format. It is assumed that the nodes undergoing a shift for a given path are also shifted in all subsequent paths, and need not be specified more than once. This reduces the amount of user input required to specify the node shifts. **To reduce the programming effort, the number of entries must be the same in all columns (paths) in the ".csv" input file, therefore, enter 0 in place of blank node numbers.**

   Input format: comma separated matrix of node numbers $[N_{ij}]$, i^{th} node in j^{th} path.

The user should exercise caution when saving the above files, especially the nodal-displacements comma separated ".csv" file to ensure a sufficient number of significant figures are saved. During the development phase, it was noticed that Excel often displayed the displacement results in scientific notation using only two decimal places. If left unchanged, only the displayed significant figures would be saved into the ".csv" file, and the lost significant figures cannot be retrieved.

Initially, the program was developed using counter-clockwise (CCW) node numbering, consistent with most finite element formulations. However, positive crack opening displacements could

only be obtained by reversing the element normals such that nodes are numbered in a clockwise (CW) fashion. Therefore, CW node numbering is used in this program. It is interesting to note that in the absence of contact definitions to prevent negative crack opening displacements, using CCW node numbering and negative displacements results in positive and accurate estimates of the SERR, which highlights the importance of first checking the deformed shape to ensure physically admissible deformations.

3 Brief Overview of Alternative Techniques

The method developed by deLorenzi to compute SIF uses a continuum mechanics approach to apply the VCET. Under elastic conditions, the approach is analytically equivalent to the J-integral (using Green's theorem to convert the path integration to an equivalent area integral).

The simplest means of extracting SIFs from finite elements results is through the so-called crack-tip opening displacement (CTOD) method. From linear-elastic fracture mechanics, the stress intensity factors are used to describe the stress and displacement fields around the crack-tip. Therefore, knowing the crack-tip displacement field from finite element results, one can compute the stress intensity factors due to the applied load. Using two-dimensional four-node quadrilateral element configuration shown in Figure 1, the stress intensity factors from this technique are given by Equation 4.

$$\begin{aligned} K_I &= \frac{2\mu\sqrt{2\pi}}{\sqrt{L_e}} \frac{v_b - v_a}{\kappa} \\ K_{II} &= \frac{2\mu\sqrt{2\pi}}{\sqrt{L_e}} \frac{u_b - u_a}{\kappa} \end{aligned} \tag{4}$$

where,

u and v = axial and lateral displacements, respectively, of the crack-tip nodes shown in Figure 1,
L_e = length of the element,
μ = material shear modulus, and
κ = material parameter given by

$$\kappa = \begin{matrix} 3-4\nu & \quad \textit{plane stress} \\ \dfrac{3-\nu}{1+\nu} & \quad \textit{plane strain} \end{matrix}$$

It can be shown that strain energy release rate is related to the stress intensity factor through

$$G_{I,II} = \frac{K_{I,II}^2}{E^*}$$

where, E* is the effective elastic modulus given by;

$$E^* = \begin{cases} E & \textit{in plane stress} \\ \dfrac{E}{1-\nu^2} & \textit{in plane strain} \end{cases}$$

An analogous procedure is the modified crack closure integral (MCCI) developed by Rybicki and Kanninen (1977), which has proven very popular in the literature. However, it is strictly applicable to linear elastic conditions. In the MCCI, the SERR is estimated by the energy required to close the crack by a small amount Δa. In validating the results of ***vcet_martec.m*** results, comparisons are made to MCCI results. According to the MCCI technique, the SERR in modes I and II are given by

$$G_I = \frac{F_y^a}{2L_e}\left(v_b - v_c\right)$$

$$G_{II} = \frac{F_x^a}{2L_e}\left(u_b - u_c\right)$$

where,

F_x^a and F_y^a = crack-tip nodal sliding and opening forces, respectively, and

u and v = sliding and opening displacements, respectively, of the crack-tip nodes shown in Figure 1.

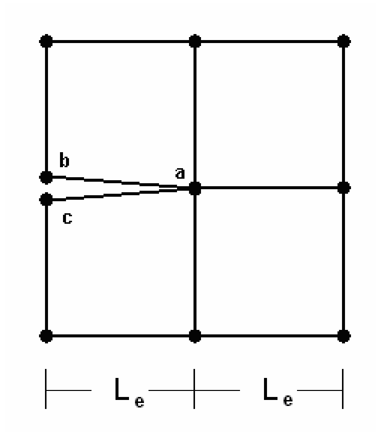

Figure 1. Finite element mesh around the crack-tip using four-node quadrilateral elements for CTOD and MCCI calculations.

The main advantage of the CTOD and MCCI over the VCET method lies in the fact the strain energy release rate components are determined in each mode separately, whereas the VCET, similar to the J-integral, only gives estimates of the total SERR. To separate the contributions of modes I and II, Bui (1983) proposed a procedure to separate modes I and II contributions to the J-integral (which can also be applied to the VCET) by separating the displacement, stress and strain fields into a symmetric (mode I) and an anti-symmetric (mode II) components.

Typically, the VCET and J-integral methods are more accurate than the MCCI, and all are far more accurate than the CTOD method. In fact, the CTOD can only produce acceptable results using singular eight-node elements. However, the computational effort required for programming the VCET or J-integral methods for an arbitrary crack under mixed-mode conditions is substantially greater than that required for the MCCI method.

3.1 Possible Sources of Error in calcj.m Program

Unfortunately, we could not run the Matlab code provided by the client (*calcj.m*), and the error could not be readily resolved. However, three possible sources of error became apparent by inspecting the structure of the *calcj.m* code, and they are:

1. The strain energy density is computed as half the product of the stress and strain tensors. This approach would result in incorrect estimates of the strain energy density since

   $$W = \frac{1}{2}\{\sigma\}^T\{\varepsilon\} = \frac{\sigma_x\varepsilon_x + \sigma_y\varepsilon_y + \tau_{xy}\gamma_{xy}}{2}$$

   whereas the tensor product yields the incorrect value given below, since the shear stress component is accounted for twice.

   $$incorrect\ W = \begin{bmatrix} \sigma_x & \tau_{xy} \\ \tau_{xy} & \sigma_y \end{bmatrix}\begin{bmatrix} \varepsilon_x & \gamma_{xy} \\ \gamma_{yx} & \varepsilon_y \end{bmatrix} = \frac{\sigma_x\varepsilon_x + \sigma_y\varepsilon_y + 2\tau_{xy}\gamma_{xy}}{2}$$

   LS-Dyna output provides the true strain, γ_{xy}, which is twice the engineering strain, ε_{xy}. Therefore, the strain matrix used in "*calcj.m*" must take this into account by modifying the LS-Dyna output results before use in the SERR calculations.

2. The correct weight function for one-point Gauss Quadrature integration is 2.0, and not 1.0 as used in "*calcj.m*" code.

3. The correspondence between the element numbers in "elcont" and those of "csig" and "ceps" is not readily apparent. Element stresses referenced in the range in the file "elcont" number between 5027 and 5527, however, only 4950 element stress and strain tensors were provided in files "csig" and "ceps".

4 Verification Examples:

4.1 Example 1

The SERR is evaluated for a centrally cracked 10 mm square plate. The plate thickness is 0.1 mm, and the crack length is 0.6 mm. Young's modulus and the Poisson ratio of the plate are 207 GPa and 0.3, respectively. According to linear elastic fracture mechanics, the exact SERR is given by

$$G_{exact} = \left(\frac{1 - 0.5\left(a/b \right) + 0.325\left(a/b \right)^2}{\sqrt{1 - \left(a/b \right)}} \right)^2 \sigma^2 \pi a$$

where,

a = half crack length,
b = half plate width, and
s = remote tensile stress,

The model is double-symmetric, therefore only a quarter of the plate is modeled. The finite element mesh used in this example is relatively coarse (558 nodes) with a focused circular 15x16 mesh around the crack-tip as shown in Figure 2. Virtual crack extension is introduced in seventeen paths, the first contains only the crack-tip node, and each successive path contains a layer of nodes surrounding the crack-tip. A truncated version of the LS-Dyna input file is given in Annex A, and the full LS-Dyna input is included in the electronic appendix (*circularElements_05.dyn*).

As illustrated in the truncated version of the input file, the elements surrounding the crack are defined under a separate part ID, and the nodes attached to these elements are numbered 1 through 256. Nodes outside this component have numbers greater than 1000. Similarly, elements surrounding the crack are numbered 1 through 240, whereas elements outside the crack region are numbered 1241 and higher.

Electronic copies of all input files used in this model are provided in electronic format in the *vcet_martec.zip* file. The input command line for this model is given by

c_nodes.csv, c_elements.csv, c_disp.csv, c_paths.csv, 12, 1, 1, 207000, 0.3, 0.001

In the first verification example used, the average SERR from all fourteen paths that include more than 1 node is -1.44 %, whereas the error in the MCCI result is 2.85 %. The CTOD cannot be expected to yield practically usable estimates of the SERR due to the use of first order elements (which cannot capture the singular displacement field). The error in the CTOD SERR is 247 %. The results are summarized in Table 1 for all paths, including one where all nodes are shifted which results in zero SERR (serves to validate the integrity of the model).

Lastly, the ***vect_martec.m*** program is used to determine the SERR in two models using triangular elements (shown in Figure 3) and a very coarse mesh (shown in Figure 4). The results from the finite element model are summarized in Table 1, which illustrates a common feature of energy based methods of extracting SERR from FE models in that an overly fine mesh is not necessary to obtain practical results. Furthermore, the applicability of the model is verified for both quadrilateral and triangular elements.

4.2 Example 2

4.2.1 Objectives

i) Verify the applicability of ***vcet_martec.m*** in finite element models using a rectangular elements of quadrilateral elements, and
ii) Verify the accuracy of the ***vcet_martec.m*** program in instances where the crack plane does not coincide with a plane of symmetry.

A 10-mm square steel plate with a central crack is analyzed using LS-Dyna, and the strain energy release rates (SERR) are determined using the virtual crack extension technique (VCET) as implemented in the ***vcet_martec.m*** code. All files required to run this problem are included in the attached *rectangular_mesh.zip* file. The input parameters are given by:

rect_nodes.csv, rect_elements.csv, rect_disp.csv, rect_paths.csv, 12, 0, 1, 207000, 0.3, 0.001

This is the same geometry and load conditions used in the earlier verification example, except the mesh consists entirely of quadrilateral elements in the crack-tip region as shown in Figure 5. Furthermore, unlike in the previous example, this model does not take advantage of symmetry about x-axis (note, the *input_parameters.dat* control card number six is set to zero as given above) since the crack-tip is not defined on a plane of symmetry (as shown in Figure 6). The SERR obtained from nine paths are summarized in Table 2, where in the ninth path all nodes are shifted by an equal amount to ensure the resulting SERR is zero in such instances. As expected, least accurate estimates of SERR are obtained when the crack is virtually extended by the shifting of only a single node.

4.3 Element Thickness Effects

The formulation of the SERR given in ***vcet_martec.m*** is in fact independent of the element thickness, assuming the elements in the crack component are of the same thickness. It was noted that the *calcj.m* code incorrectly divides the result of the SERR integral by (Δa x t), when in fact it should only be divided by (Δa) for two-dimensional plane stress and plane strain elements. The thickness effects are in fact cancelled out by performing the volume integration as given below

$$\frac{\int_V I\ dV}{\Delta a\ t} = \frac{\int_A I\ t\ dA}{\Delta a\ t} \xrightarrow[\text{thickness}]{\text{constant}} = \frac{\int_A I\ dA}{\Delta a}$$

The expressions given by deLorenzi (1985) also reflect this observation. Specifically, equation 21 in deLorenzi's work applies to plane stress and plane strain elements, regardless of the element thickness, as long as all elements (in the integral) share the same thickness.

However, it was noted that LS-Dyna pressure input (using *LOAD_SEGMENT) only produced the correct displacement magnitudes when a unit thickness is used in *SECTION_SHELL definition. Therefore, all verification examples were developed using a unit-thickness plane stress elements. For elements with different thickness values, the program ***vcet_martec.m*** would still produce the correct results, given that the nodal displacement results from LS-Dyna are correctly obtained. This observation (most likely) does not apply to models when the load is applied through nodal loading.

For example, in the attached *rectMesh_06.dyn* model (included in *rectangular_mesh.zip*), changing the element thickness under *SECTION_SHELL from 1.0 to 0.1 would result in a ten-fold increase in displacements. This was not expected, since the load is applied as uniform pressure using the control cards shown under *LOAD_SEGMENT. It was expected that LS-Dyna would factor in the element thickness effects when the pressure load is converted to an equivalent nodal point load.

Table 1. SERR Estimates from Example 1 FEA Models

	SERR (N/mm)		
Path	Circular mesh	Triangular mesh	Coarse mesh
1	8.2896	8.7604	8.2670
2	9.1198	9.3749	10.0396
3	9.2698	9.2785	0
4	9.3268	9.2442	
5	9.3607	9.2368	
6	9.3823	9.2401	
7	9.3978	9.2434	
8	9.4078	9.2459	
9	9.4165	0	
10	9.4271		
11	9.4274		
12	9.4343		
13	9.4378		
14	9.4387		
15	9.4448		
16	0		
Average	9.305413	9.203007	9.153285
MCCI	9.5081	9.2827	10.3381
CTOD	31.1626	22.0386	53.1984
E (GPa)	207		
Exact	9.24870		

Table 2. Summary of Example 2 Results.

Path	SERR (N/mm)
1	7.4031
2	8.9554
3	9.1995
4	9.2632
5	9.2747
6	9.2896
7	9.2867
8	9.2858
9	0
Average (paths 1-8): 8.99475 N/mm	
Average (paths 2-8): 9.22129 N/mm	
Exact = 9.24870 N/mm	

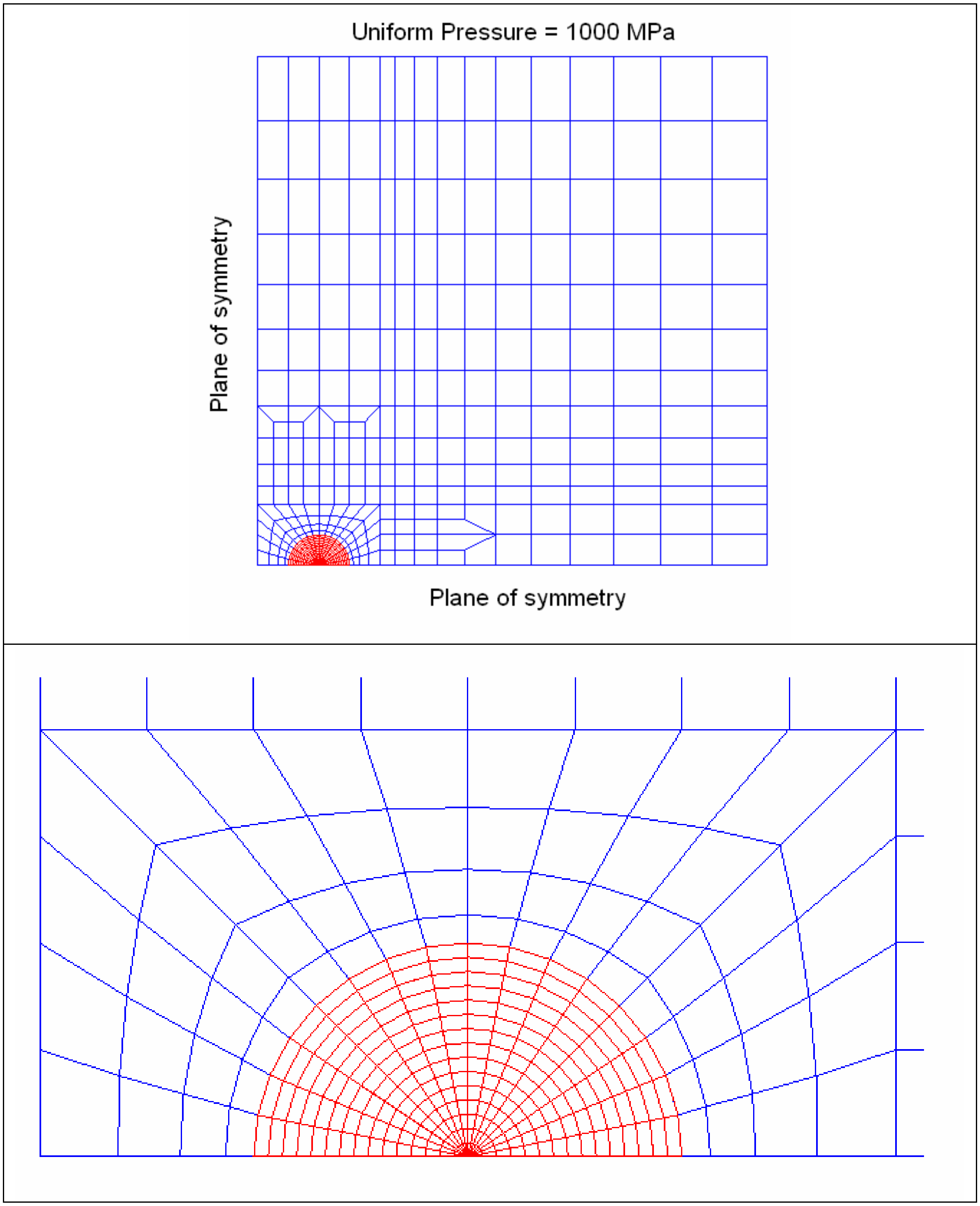

Figure 2. Finite element model using circular mesh around the crack-tip (Example 1).

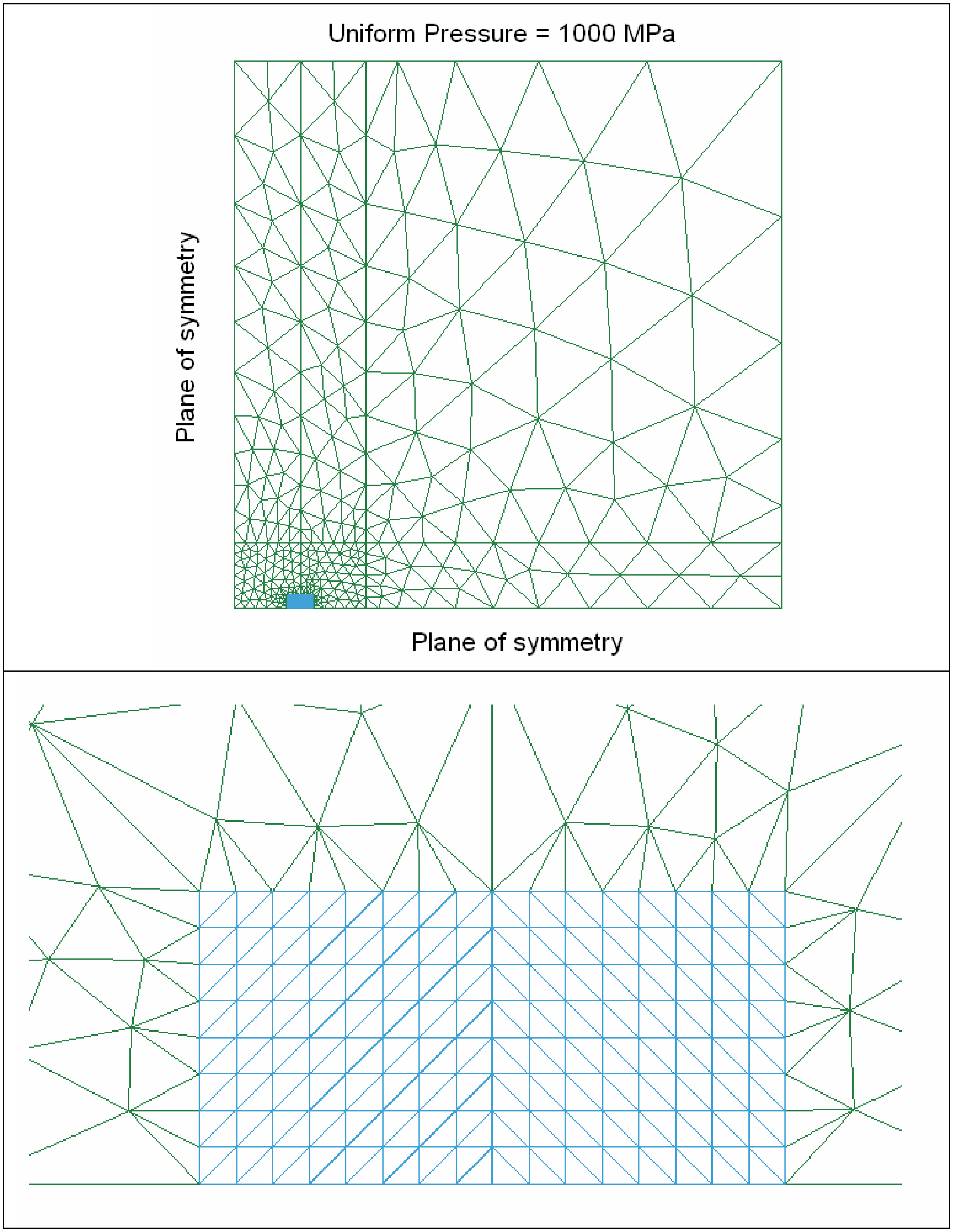

Figure 3. Finite element model using triangular elements around the crack-tip (Example 1).

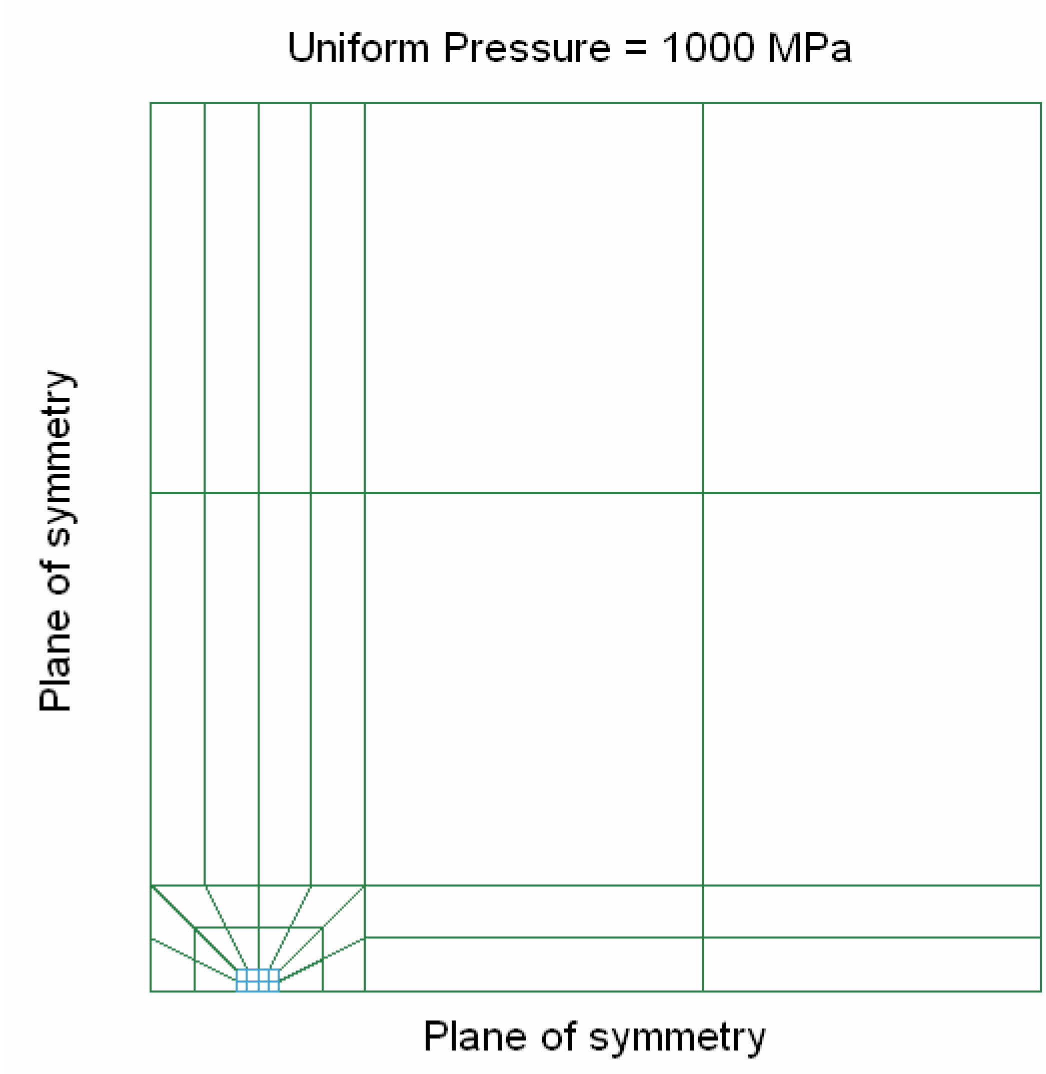

Figure 4. Finite element model using a coarse mesh (Example 1).

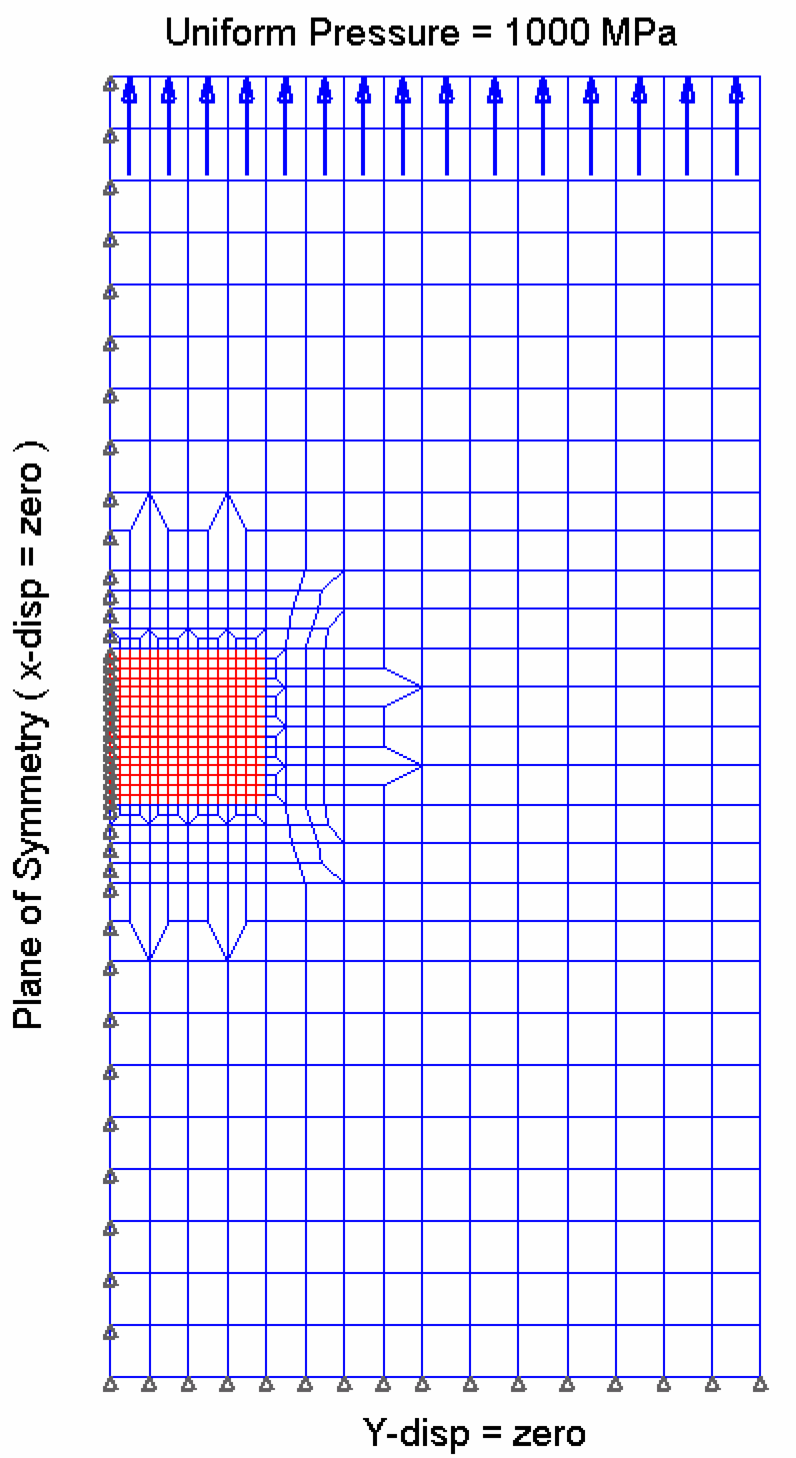

Figure 5. Finite element model using a rectangular mesh around the crack-tip (Example 2).

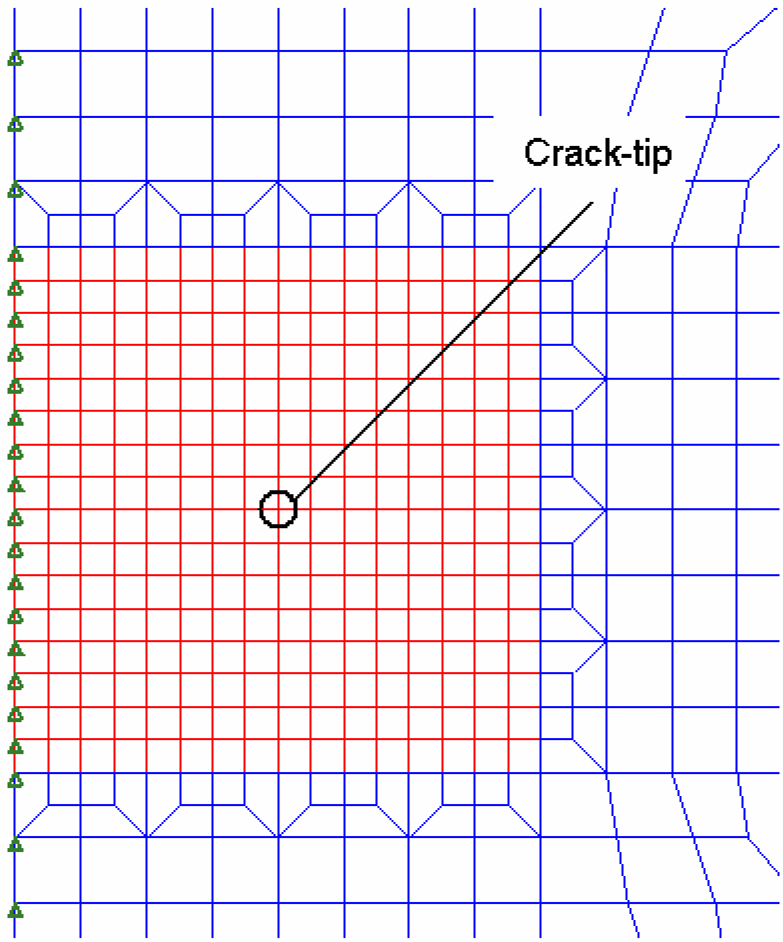

Figure 6. Close-up of elements surrounding the crack-tip (Example 2).

References

HD Bui, 1983, Journal of the Mechanics and Physics of Solids, 1983, 439-448

HG Delorenzi, 1985, Engineering Fracture Mechanics, (21) 129-143.

EF Rybicki, and MF Kanninen, 1977, Engineering Fracture Mechanics (9) 931-938.

This page intentionally left blank.

Annex A Sample Input Fle

```
*KEYWORD
*TITLE
Circular Crack Mesh
$
$       Solution parameters not shown
$
*NODE
    1 0.5260634364773 0.0305469888013          0.0
$
$       Nodes in the CrackElements component
$       256 nodes in total
$
  256         0.58          0.0          0.0
 1231 0.2674344206075  0.461001262414          0.0
$
$       Nodes in the RestOfElements component
$       302 nodes in total
$
 1580 2.0322314049587 2.7636363636364          0.0
$
$       Material definition
$
*MAT_ELASTIC
     1 7.900E-09  207000.0      0.3
$
$       Definition of element parts (components)
$
*PART
$
$       Part ID 2 contains the CrackElements component
$
     2       1       1
$
$       Part ID 3 contains the RestOfElements component
$
     3       1       1
*SECTION_SHELL
     1      12
   0.1     0.1     0.1     0.1
*ELEMENT_SHELL
   2     2   194     12     26     26
  17     2   194     26     40     40
$
$       Triangular elements at the crack-tip, 12 elements in total
```

```
$       Note repeated node definition for nodes 3 and 4
$
   211     2    194    242    228    228
   226     2    194    256    242    242
*ELEMENT_SHELL
     1     2     26     12     13     27
$
$       Quadrilateral elements in the CrackElements component
$       224 elements in total
$       Therefore, total number of elements in the CrackElements
$       component is 240 elements
$
   240     2    243    209    216    229
  1241     3   1231   1249   1260   1261
$
$       Elements in RestOfElements component.
$
  1519     3   1580   1572   1558   1559
$$
$$ Sets Defined In HyperMesh
$$
*SET_NODE_LIST
         1
       194
*SET_NODE_LIST
         2
        12        26        40        54        68        82        96       110
       124       138       165       166       193       195       228       242
       256
$
$       Total of 16 node sets for the paths defining the crack extension
$
*SET_NODE_LIST
        16
       209       210       211       212       213       214       215       216
       217       218       219       220       221       222       223       224
       225
*SET_NODE_LIST
       101
$
$       Contains nodes 1 to 256 (defined in 16 sets above) in one set ... this set
$       is used to save node displacement results required for fracture analysis.
$
*DATABASE_HISTORY_NODE_SET
       101

*END
```

Distribution list

Document No.: DRDC Atlantic CR 2008-274

LIST PART 1: Internal Distribution by Centre

2	DRDC Atlantic DLP (Attn: Christopher Bayley) (1 Hardcopy, 1 CD)
1	DRDC Atlantic TAG Leader (Attn: N. Pegg)
5	DRDC Atlantic Library (1 Hardcopy, 4 CDs)
8	TOTAL LIST PART 1

LIST PART 2: External Distribution by DRDKIM

1	Library and Archives Canada
1	DRDKIM
1	Martec Limited 1888 Brunswick Street, Suite 400 Halifax, Nova Scotia Canada B3J 3J8
3	TOTAL LIST PART 2

11 TOTAL COPIES REQUIRED

This page intentionally left blank.

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (The name and address of the organization preparing the document. Organizations for whom the document was prepared, e.g. Centre sponsoring a contractor's report, or tasking agency, are entered in section 8.)

   Martec Limited
   1888 Brunswick Street, Suite 400
   Halifax, Nova Scotia
   Canada B3J 3J8

2. SECURITY CLASSIFICATION (Overall security classification of the document including special warning terms if applicable.)

   UNCLASSIFIED

3. TITLE (The complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S, C or U) in parentheses after the title.)

   Virtual Crack Extension Technique

4. AUTHORS (last name, followed by initials – ranks, titles, etc. not to be used)

   Shahin K., ; Koko T.S.

5. DATE OF PUBLICATION (Month and year of publication of document.)

   December 2008

6a. NO. OF PAGES (Total containing information, including Annexes, Appendices, etc.)

   36

6b. NO. OF REFS (Total cited in document.)

   3

7. DESCRIPTIVE NOTES (The category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered.)

   Contract Report

8. SPONSORING ACTIVITY (The name of the department project office or laboratory sponsoring the research and development – include address.)

   Defence R&D Canada – Atlantic
   9 Grove Street
   P.O. Box 1012
   Dartmouth, Nova Scotia B2Y 3Z7

9a. PROJECT OR GRANT NO. (If appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant.)

   11gu05

9b. CONTRACT NO. (If appropriate, the applicable number under which the document was written.)

   W7707-088096/A

10a. ORIGINATOR'S DOCUMENT NUMBER (The official document number by which the document is identified by the originating activity. This number must be unique to this document.)

   08.410438

10b. OTHER DOCUMENT NO(s). (Any other numbers which may be assigned this document either by the originator or by the sponsor.)

   DRDC Atlantic CR 2008-274

11. DOCUMENT AVAILABILITY (Any limitations on further dissemination of the document, other than those imposed by security classification.)

   Unlimited

12. DOCUMENT ANNOUNCEMENT (Any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in (11) is possible, a wider announcement audience may be selected.))

   Unlimited

13. ABSTRACT (A brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual.)

In order to numerically calculate the strain energy release rate (SERR) using finite element models, an implementation of the virtual crack extension technique was developed within a Matlab script. From the nodal data read in from the ascii output files produced by LS-Dyna, the SERR was calculated from contours of elements which were positioned around the crack tip. In order to validate the implementation, a mesh sensitivity study of a center cracked panel was carried out. This study concluded that the SERR implementation was relatively independent of mesh size and shape.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (Technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus, e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus identified. If it is not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title.)

J-Integral, Fracture Mechanics, Virtual Crack Extension, FEM

This page intentionally left blank.

Defence R&D Canada

Canada's leader in defence
and National Security
Science and Technology

R & D pour la défense Canada

Chef de file au Canada en matière
de science et de technologie pour
la défense et la sécurité nationale

DEFENCE DÉFENSE

www.drdc-rddc.gc.ca